Source of Acquisition
NASA Marshall Space Flight Center

MSFC Technology Expo 2006 – Advanced Damage Tolerance Analysis of International Space Station Pressure Wall Welds

Abstract

EM20/MSFC has sponsored technology in the area of advanced damage tolerance analysis tools used to analyze the International Space Station (ISS) pressure wall welds. The ISS European modules did not receive non-destructive evaluation (NDE) inspection after proof test. In final assembly configuration, most welds could only be inspected from one side, and some welds were uninspectible. Therefore, advanced damage tolerance analysis was required to determine the critical initial flaw sizes and predicted safe life for the pressure wall welds. EM20 sponsored the development of a new finite element tools using FEA-Crack and WARP3D to solve the problem. This presentation gives a brief overview of the new analytical tools and the analysis results.

MSFC Technology Expo 2006

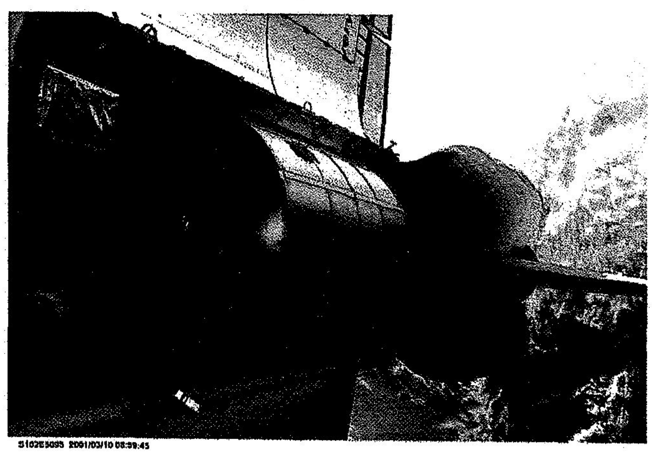
Advanced Damage Tolerance Analysis of International Space Station Pressure Wall Welds

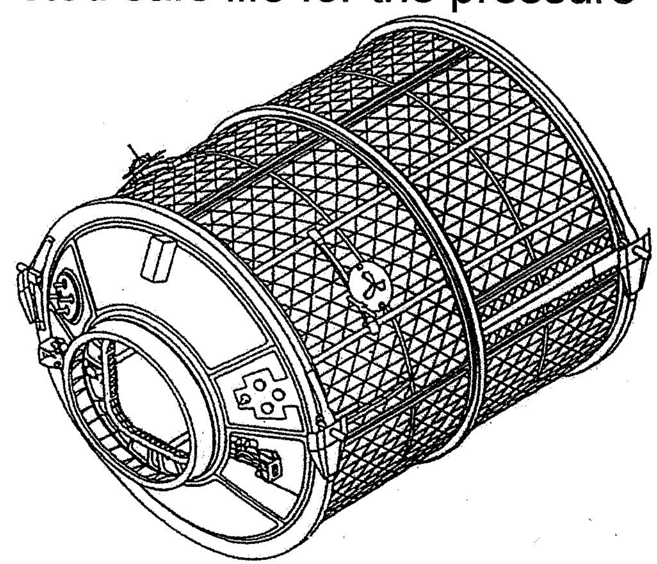
08/24/06

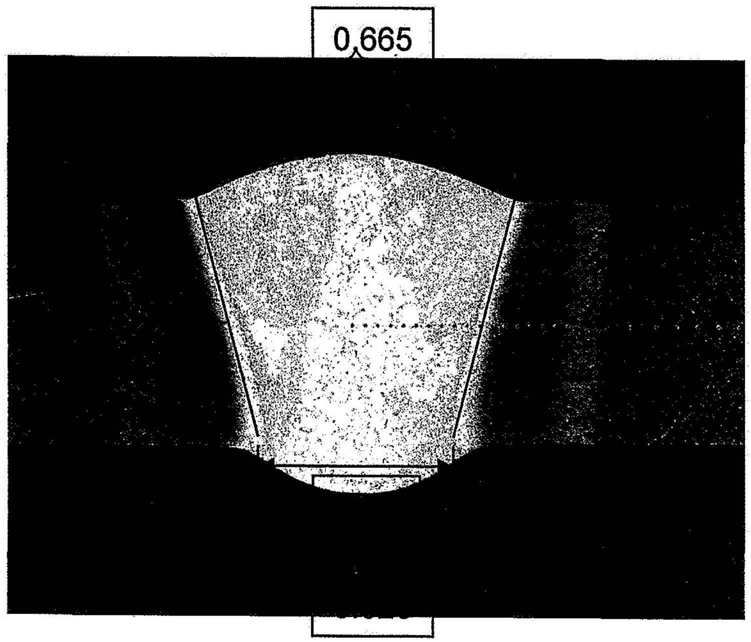
Phillip A. Allen

Damage Tolerance Assessment Branch – EM20
NASA MSFC

- Multi-Purpose Logistics Module (MPLM) – Cargo container used to transport supplies to and from the International Space Station (ISS)
- Carried in the Orbiter Payload Bay and attached to the ISS to be unloaded and reloaded while on orbit.
- <u>ISSUE</u>: Pressure shell welds did not receive non-destructive evaluation (NDE) inspection after proof test. In final assembly configuration most welds can only be inspected from one side, and some welds are uninspectible.
- Advanced damage tolerance analysis and testing was required to determine the critical initial flaw size and predicted safe life for the pressure wall welds.







Cross-Section Through Weld

KEY ANALYSIS CHALLANGES

- Large strength mismatch between weld and parent material
- Welds contain various degrees of peaking and mismatch which greatly effect weld stresses
- Weld residual stress issues
- Elastic-plastic crack growth and ductile tearing (crack instability)
- Complex test specimen geometry and boundary conditions

New Technology Developed or Utilized to Solve This Problem

- EM20 sponsored the development of a new mesh generation module for FEA-Crack that creates a mesh of a surface crack in welded plate.
- The mesh generation tool gives the user control over the size of the weld and heat-affected zone (HAZ) geometries and control over the amount of peaking and mismatch in the simulated weld.
- Analyses to determine the ductile tearing behavior of the welds were conducted using the research finite element code WARP3D using cohesive element theory.
- The development direction and development cost of both FEA-Crack and WARP3D are supported extensively by EM20/MSFC.

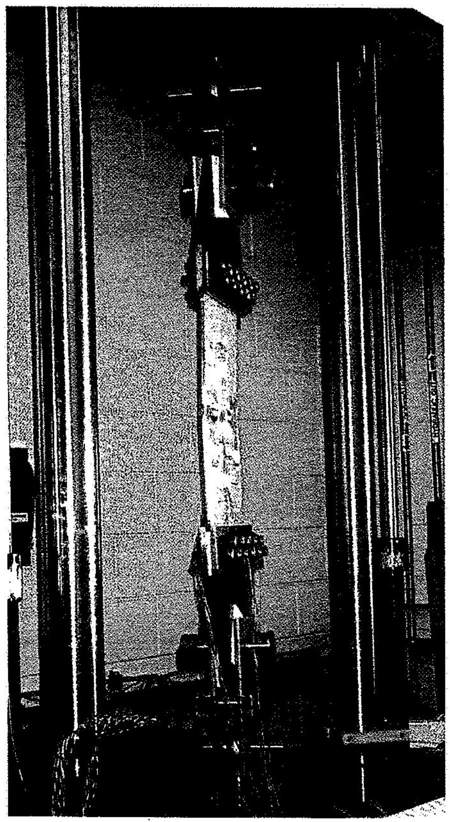
WARP3D - Release 15

3-D Dynamic Nonlinear Fracture Analysis of Solids Using Parallel Computers and Workstations

PART I

Peaking and Mismatch Specimen Analysis

Test SP36-2: Cyclic Load Test of Peaking and Mismatch Panel with Surface Crack



Test Overview

Sample: SP36-2
W = 266.7 mm (10.5 in.)
B = 7.01 mm (0.276 in.)

Peaking = 3.4 degrees
MM = 0.53 mm (0.021 in.)

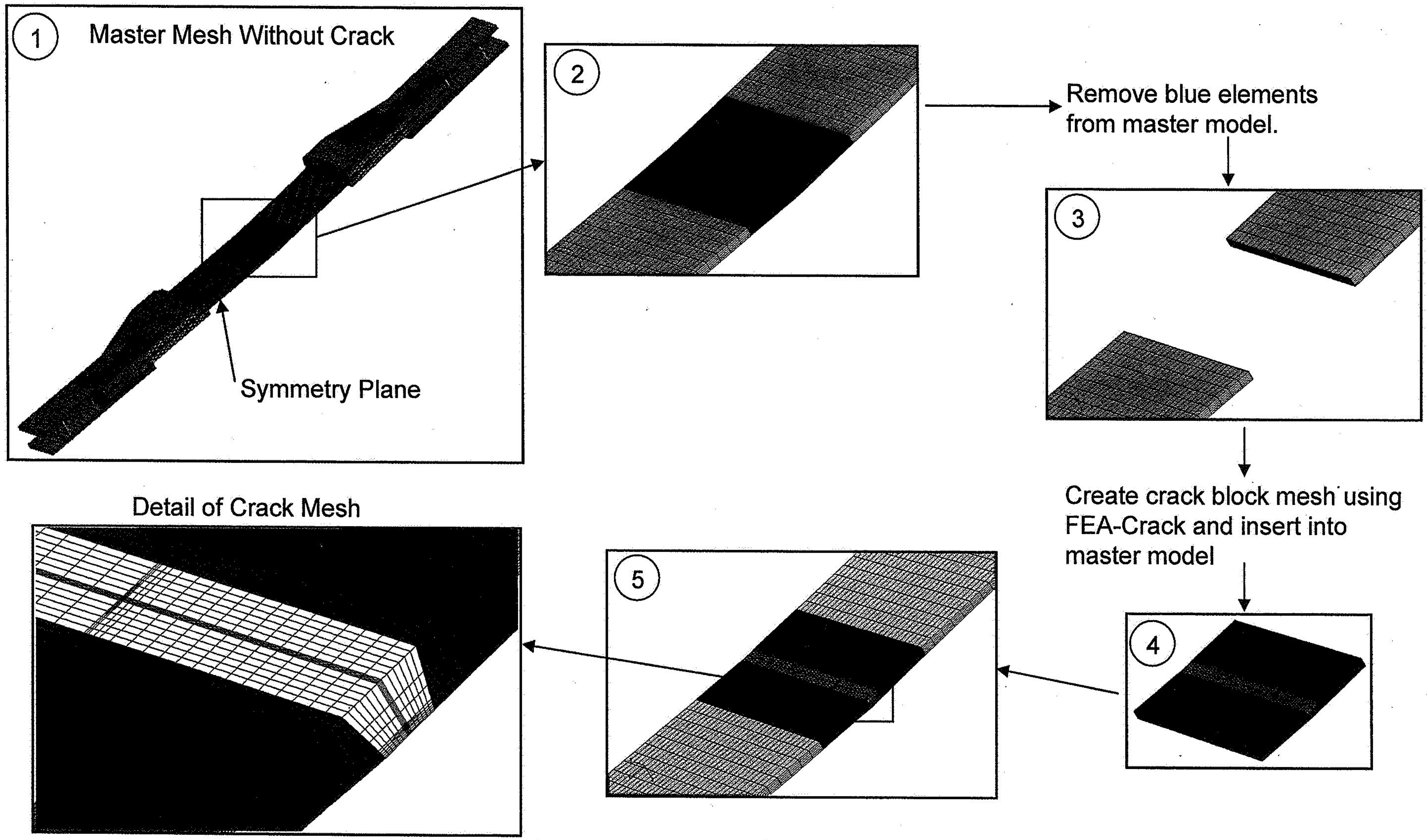
Initial Crack Size: 2c=77.7mm (3.06 in.)
a=6.25mm (0.246 in.)

Remote stress = 57.1 MPa (24.0 kip load)
Stress Ratio, R = 0.0167 (400 lbs min)

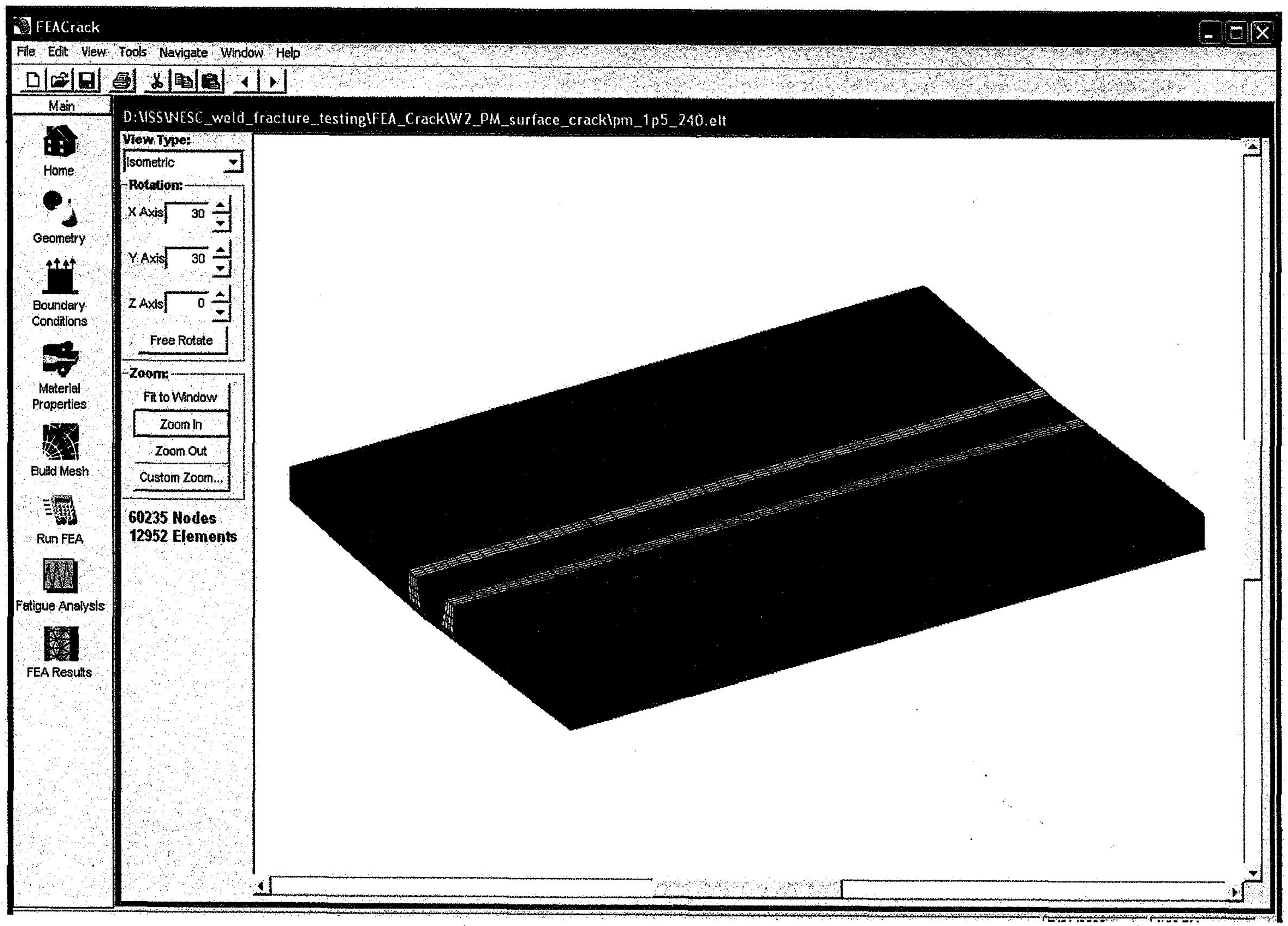
Total Cycles = 4247
Leak Occurred on cycle ≈ 80.
Failure Crack Size: 2c ≈ 146 mm (5.75 in.)

- 24 kip load produces stresses/strains in the peaking and mismatch specimen that bound the max. stresses and strains in the MPLM cone welds
- Initial surface crack fully transitions to a through crack after about 1600 cycles
- Created specimen FEM's with both surface and through cracks to investigate stress intensity values and crack opening behavior for peaking and mismatch conditions.
- Details of finite element model construction and results are given on following charts.

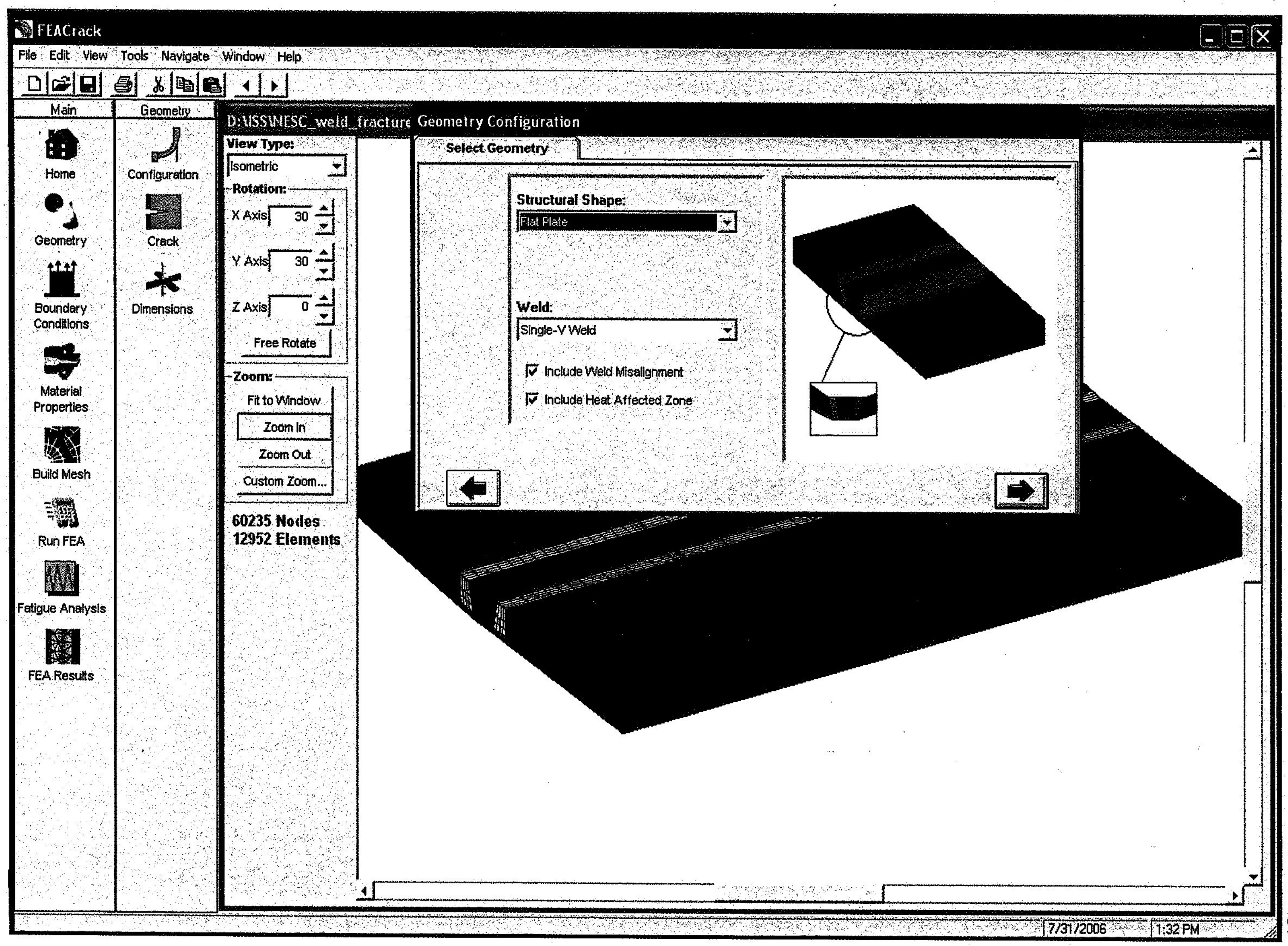
Advanced Damage Tolerance Analysis of Welded Structures



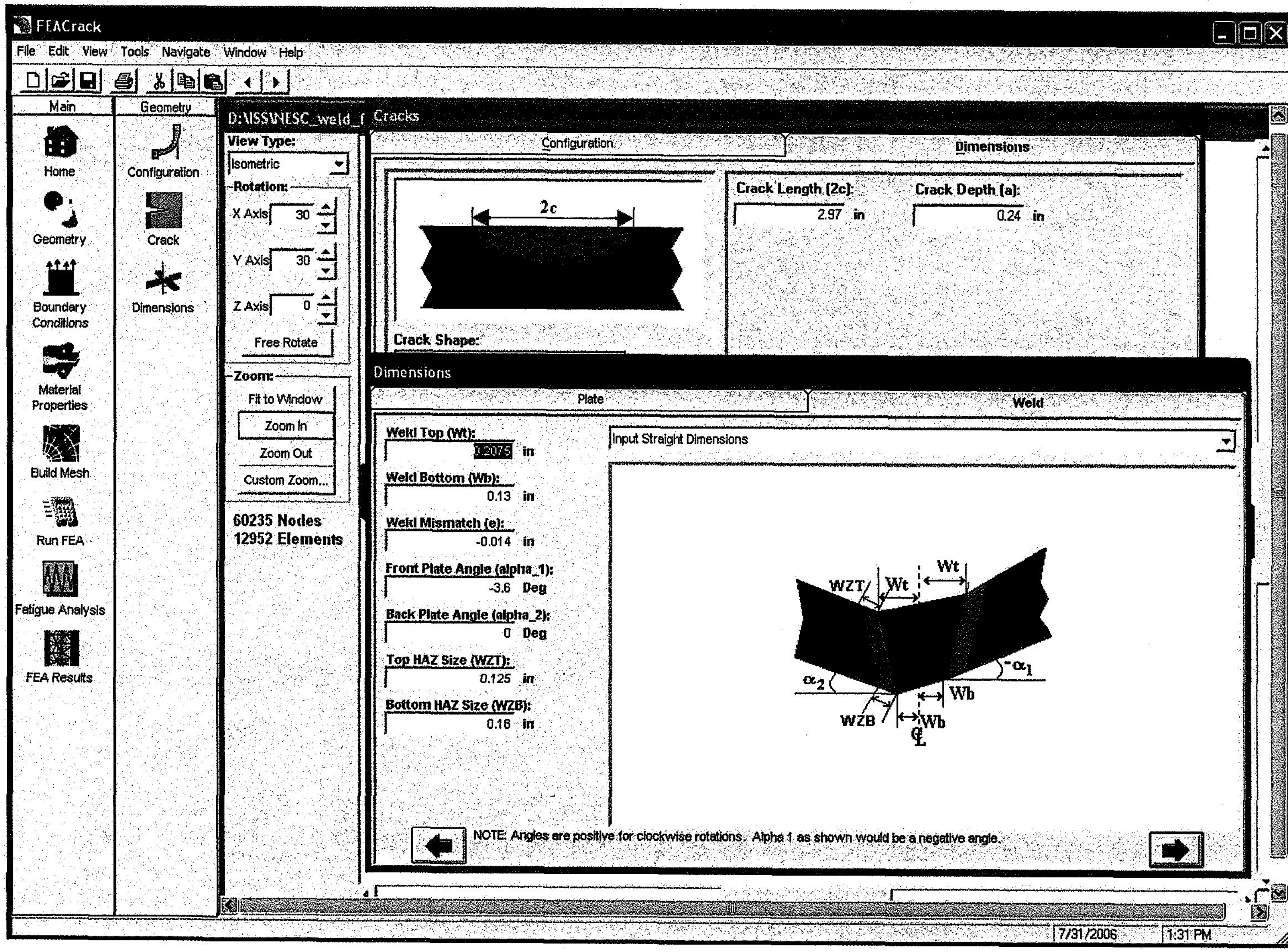
Screen Shot From FEA-Crack Mesh Generation Module



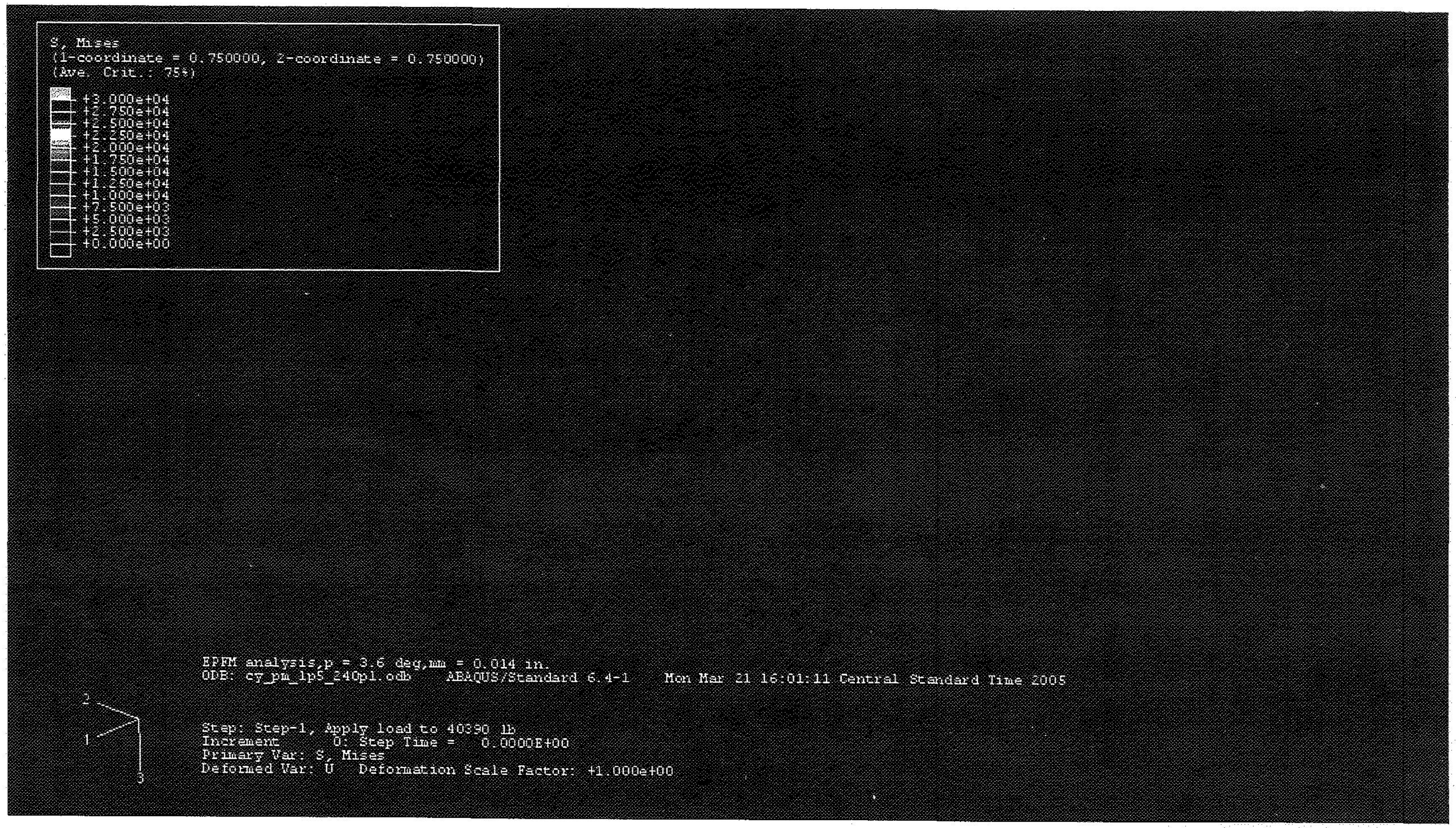
Screen Shot From FEA-Crack Mesh Generation Module



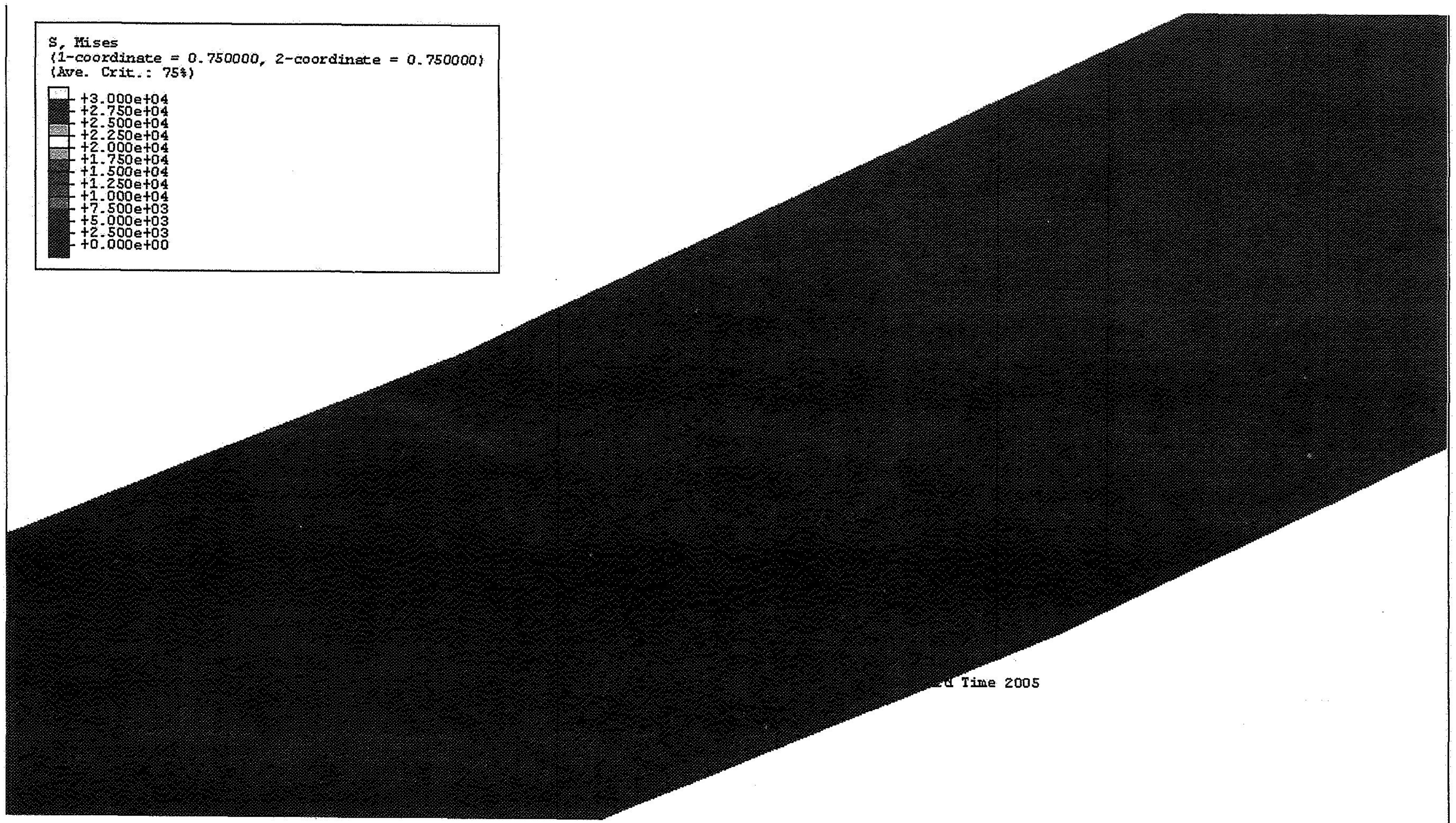
Screen Shot From FEA-Crack Mesh Generation Module



Simulation of Cyclic Test with Surface Crack, von Mises Stress

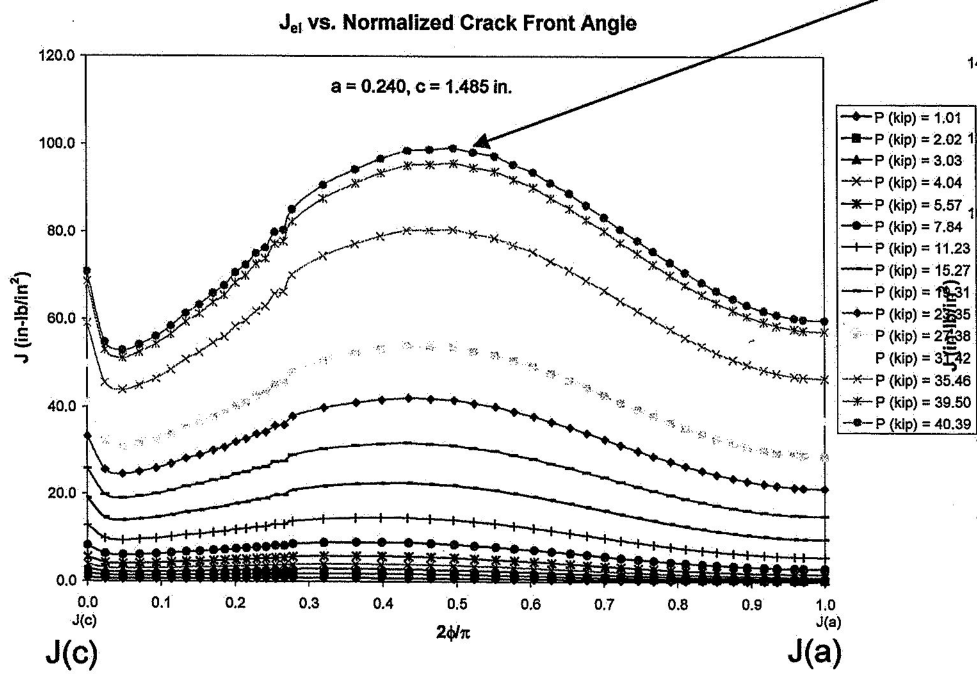


Tensile Load with Surface Crack, von Mises Stress

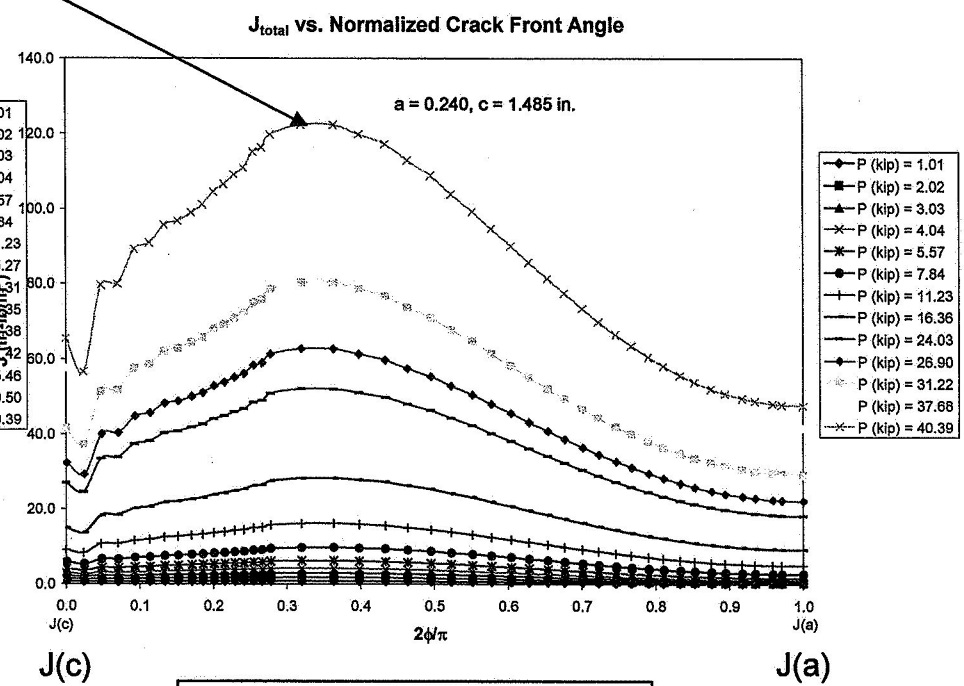


Typical Analysis Results - J vs. Normalized Crack Front Angle

NOTE: J_{max} is not at J(a) or J(c)

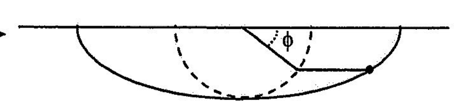


Elastic Solution

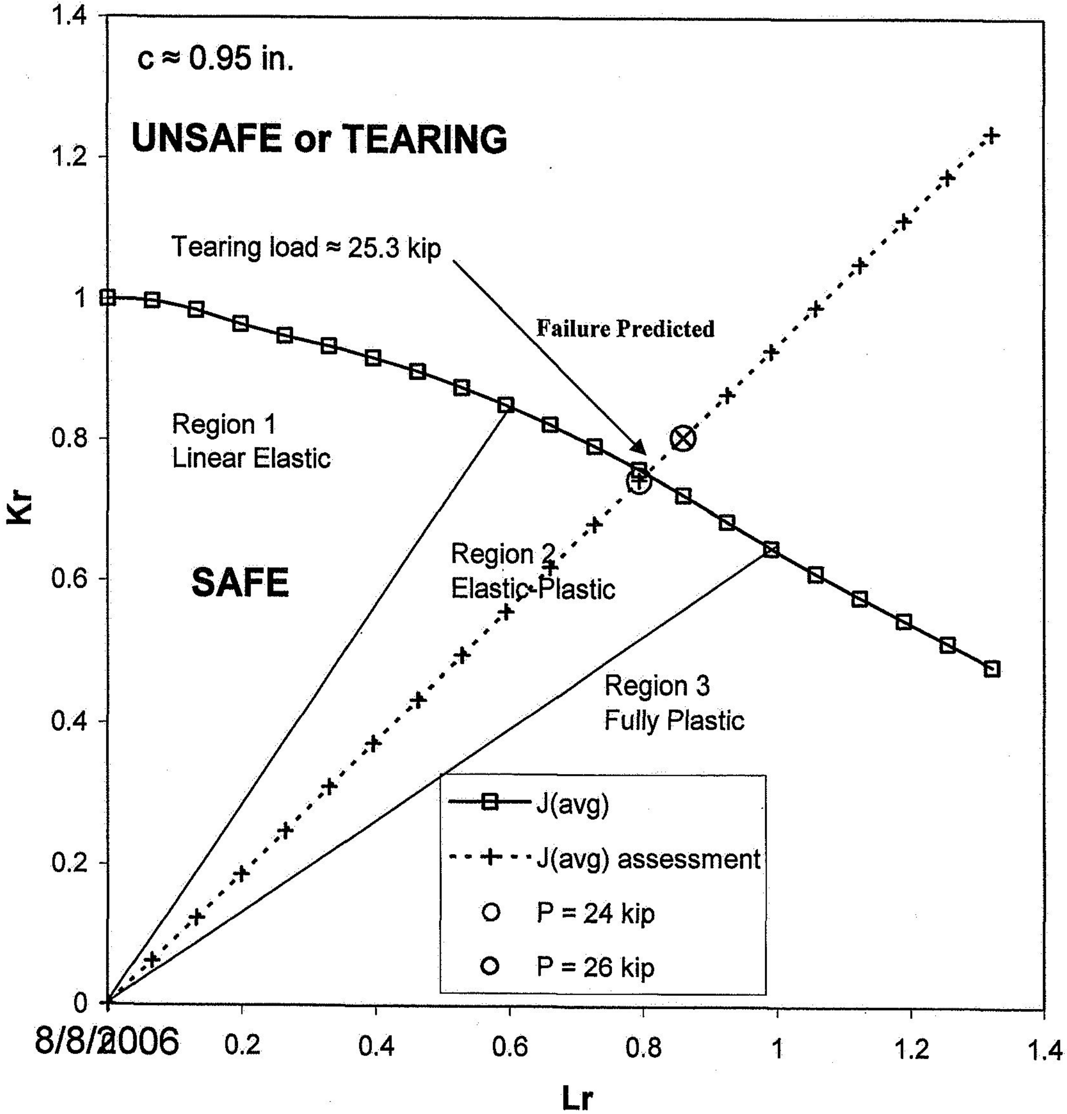


Elastic-Plastic Solution

Surface Crack Front Angle →



Failure Assessment Diagram (FAD) Results for Thru Crack c ≈ 0.950 in.



Comparison with Test Data

- Analysis assessment line for 0 to 40 kip
- Each consecutive data symbol on the assessment lines represents a 2 kip increase in load
- To create assessment lines, assumed $J_{IC} = (K_c)^2/E'$ ….($K_c \approx 40$ ksi-in$^{0.5}$)
- Analysis agrees well with tearing load of 25.3 kip

Equations to construct the FAD

$$K_r^* = \sqrt{\frac{J_{elastic}}{J_{total}}} \qquad L_r^* = \frac{P}{P_{ref}}$$

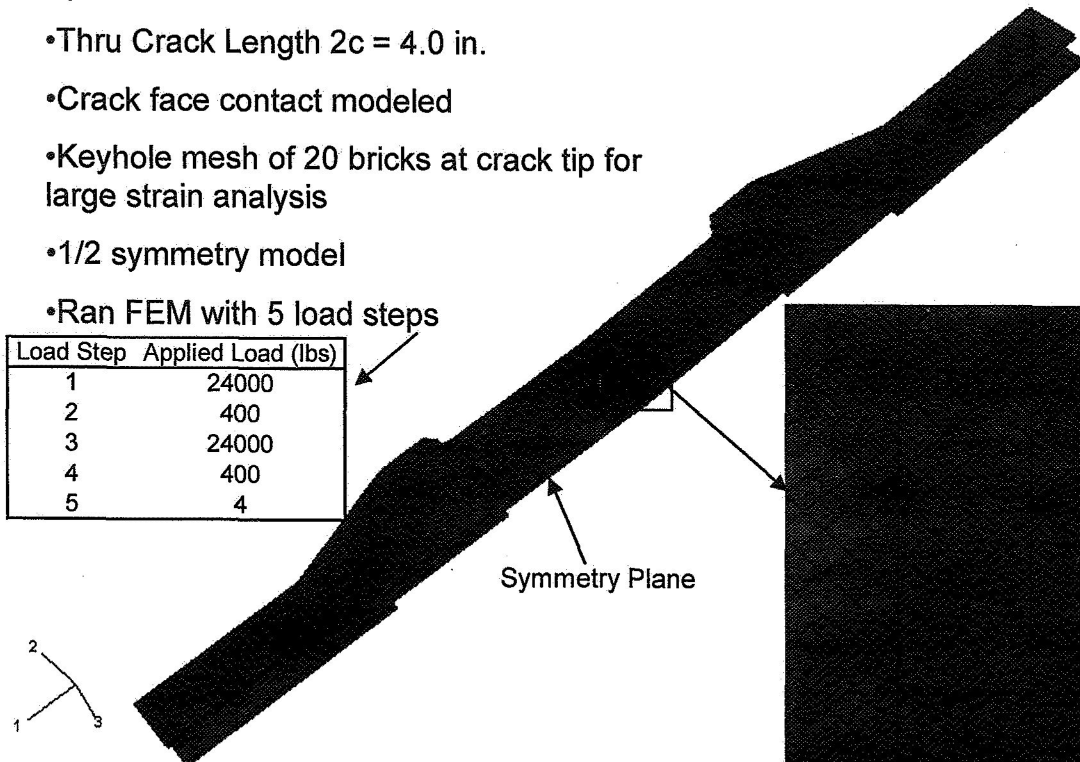
P_{ref} is the applied load at net section yielding and can be evaluated as the load that satisfies the equation below.

$$\left.\frac{J_{total}}{J_{elastic}}\right|_{P=P_{ref}} = 1 + \frac{0.002E}{\sigma_{ys}} + \frac{1}{2}\left(1 + \frac{0.002E}{\sigma_{ys}}\right)^{-1}$$

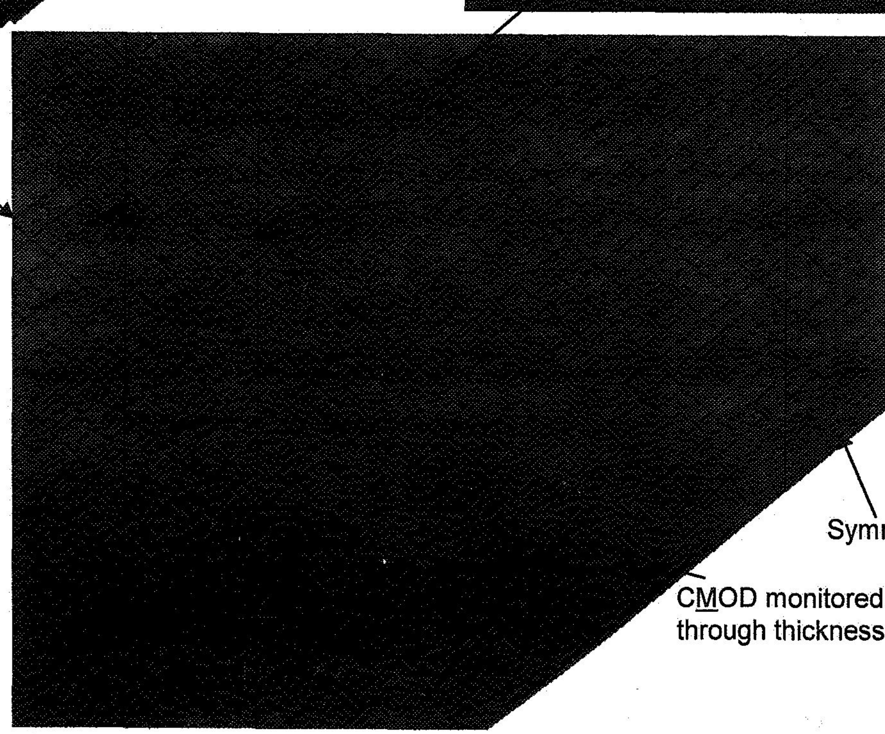
FEM of Peaking and Mismatch Specimen with Thru Crack

- ABAQUS FEM of Peaking and Mismatch Specimen 36-2
- Thru Crack Length 2c = 4.0 in.
- Crack face contact modeled
- Keyhole mesh of 20 bricks at crack tip for large strain analysis
- 1/2 symmetry model
- Ran FEM with 5 load steps

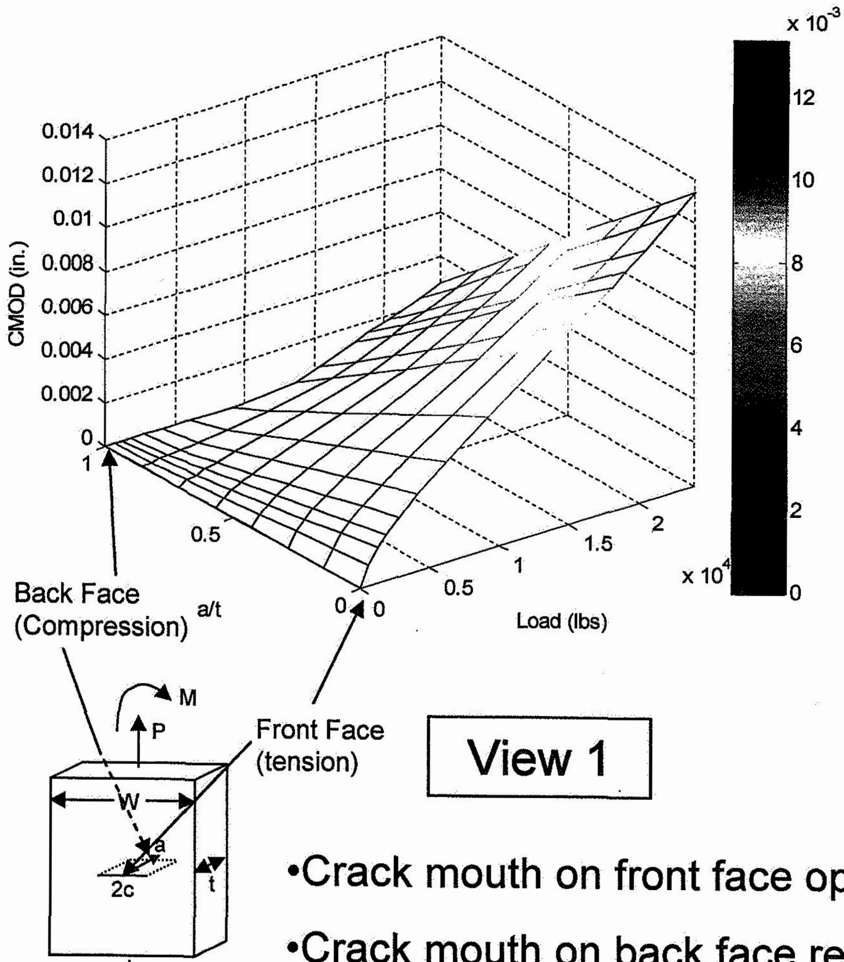
Load Step	Applied Load (lbs)
1	24000
2	400
3	24000
4	400
5	4



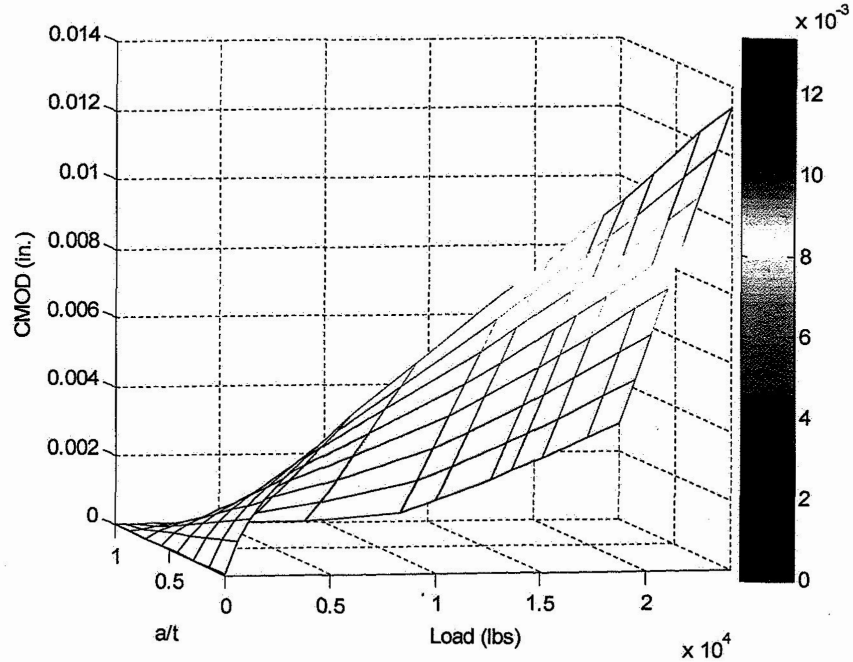
Detail of Keyhole mesh; Keyhole dia. = 0.0032 in.



2 Views of Crack Mouth Opening Analysis Results (Crack "Mouth" at center of crack length)



View 1



View 2

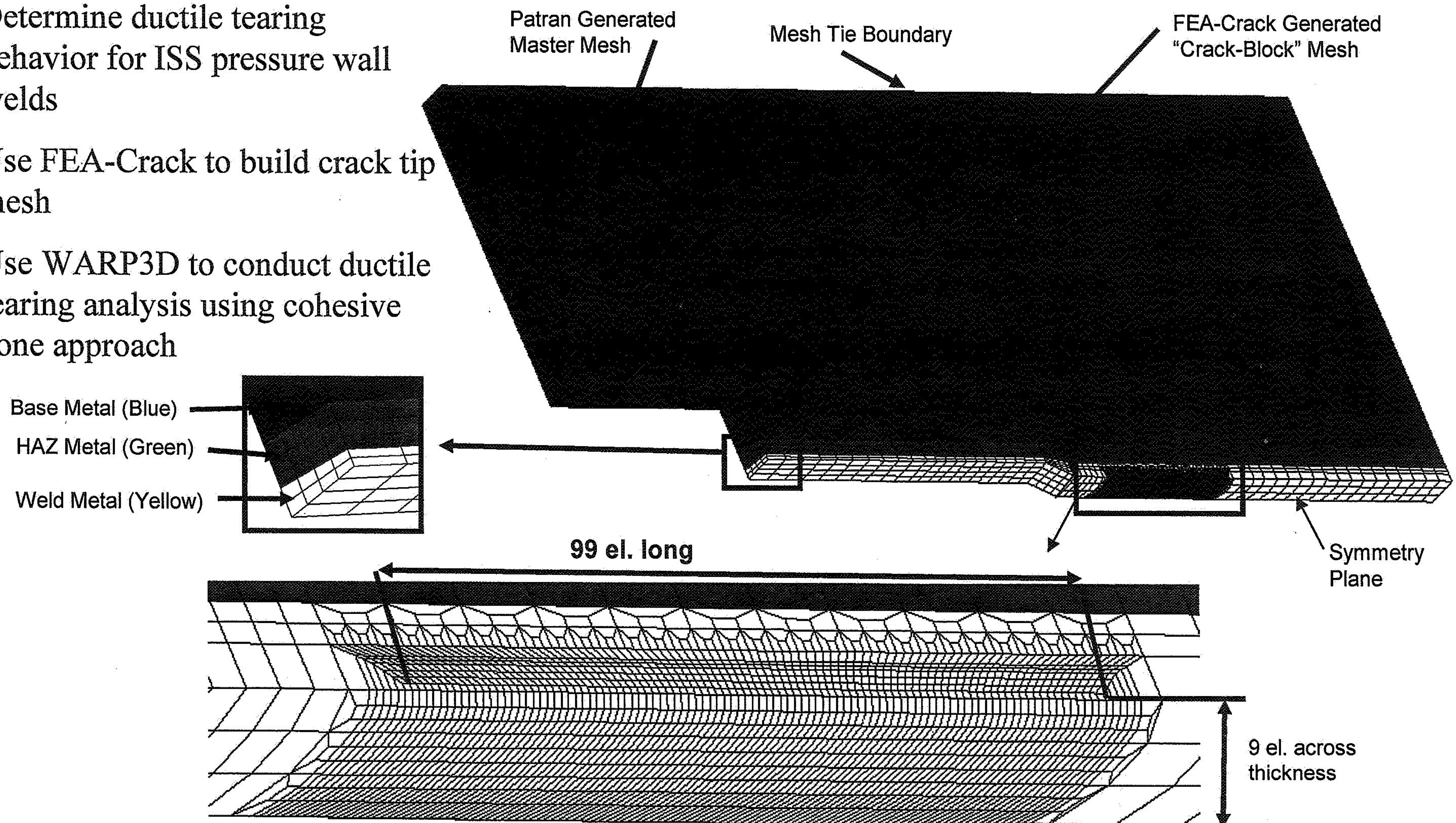
- Crack mouth on front face opens as a function of applied load
- Crack mouth on back face remains closed until load exceeds about 10,000 lbs

PART II

Compact Tension, C(T), Specimen Ductile Tearing Analysis

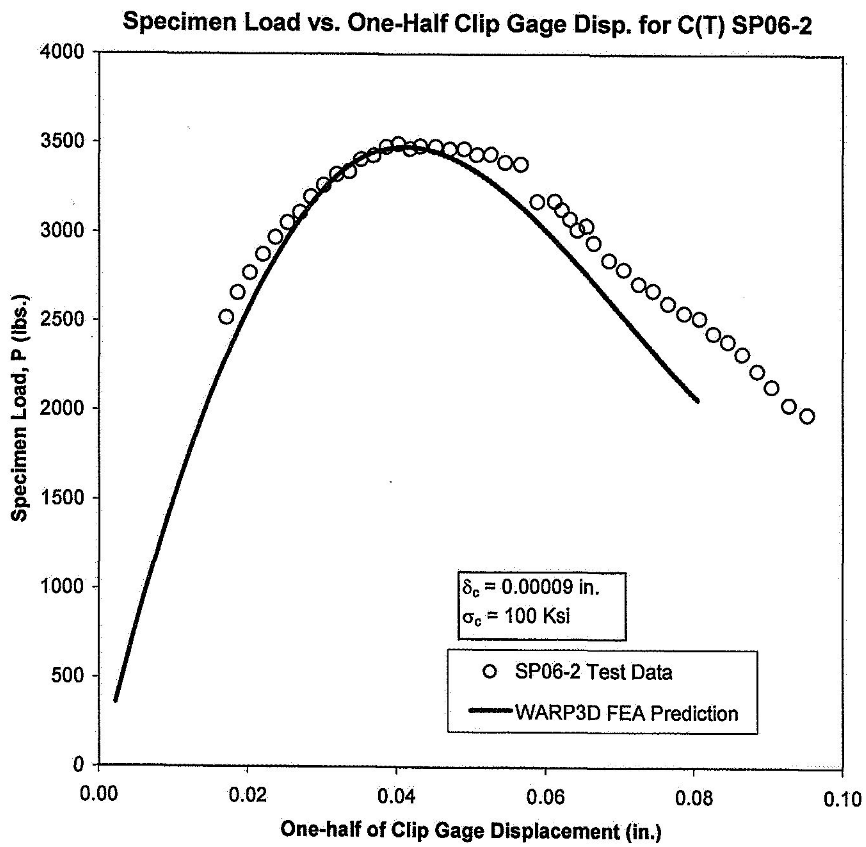
Cohesive Element FEM of C(T) Tearing Test

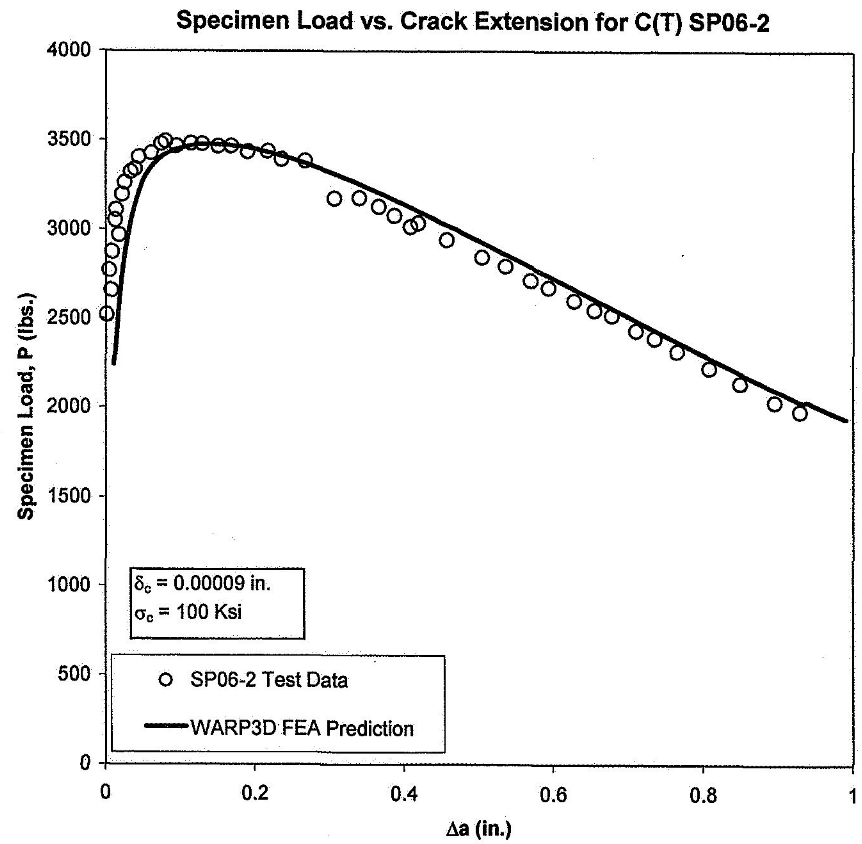
- Determine ductile tearing behavior for ISS pressure wall welds
- Use FEA-Crack to build crack tip mesh
- Use WARP3D to conduct ductile tearing analysis using cohesive zone approach



Typical Cohesive Element FEM Results

Results of Cohesive Zone Calibration of Tearing





ABAQUS Cohesive Element FEM of C(T) Ductile Tearing Test, von Mises Stress